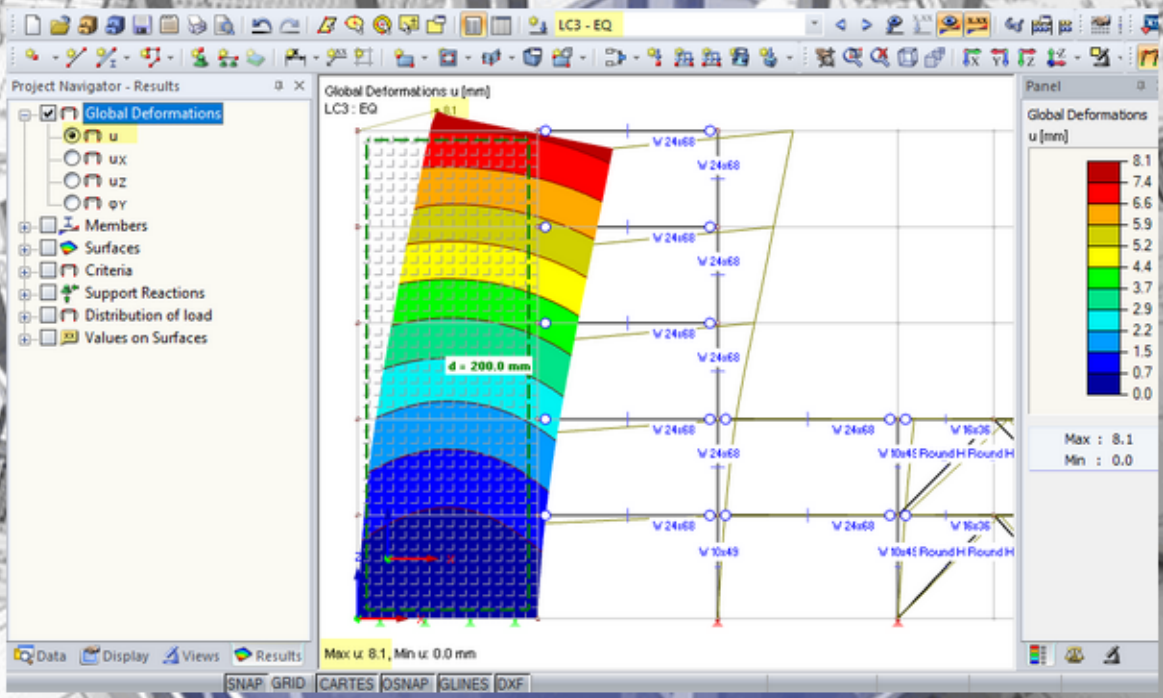
Structural Modeling and Design using RFEM



Classroom in a Book

Leonil Longno

Structural Modeling and Design using RFEM

Classroom in a Book

Copyright©2021

All rights reserved. Use of this publication (this "Work") is subject to these terms. Except as permitted under the Copyright Act of 1976, as amended, and the right to store and retrieve one copy of this Work, you may not decompile, disassemble, reverse engineer, reproduce, modify, create derivative works based upon, transmit, distribute, disseminate, sell, publish or sublicense this Work or any part without Publisher's prior written consent. You may use this Work for your own commercial and personal use; any other use of this Work is strictly prohibited. Your right to use this Work may be terminated if you fail to comply with these terms. No part of this Work may be reproduced, distributed, or transmitted in any form or by any means, including photocopying, recording, or other electronic or mechanical methods, without the prior written permission of Publisher. Written requests for permission should be addressed to: leonil@outlook.com

This work shall not be construed as a rendering of engineering or other professional advice and/or services by publisher or author. Publisher and author make no representations or warranties with respect to the accuracy or completeness of the contents of this work. Publisher and author specifically disclaim and you waive all representations and warranties (whether express, implied or statutory), including, without limitation, any warranty or condition (a) of fitness for a particular purpose, non–infringement, title, satisfactory quality, accuracy, or (b) arising from any course of dealing, course or performance, or usage in the industry. no warranty may be created or extended by sales or promotional materials. neither publisher nor author shall be liable for any damages arising from reliance on the accuracy of this work, any "construction failure" related to the use of or reliance on this work, or otherwise from the use of this work. Under no circumstances shall publisher or author be liable for any indirect, incidental, special, punitive, consequential or similar damages that result from the use of or inability to use the work, even if any of them has been advised of the possibility of such damages. This imitation of liability shall apply to any claim or cause whatsoever whether such claim or cause arises in contract, tort, or otherwise. reference or citation within this work to any organization or website does not constitute an endorsement by either published or author.

Trademarks: RFEM and RSTAB are registered trademarks of Dlubal Software GMBH
To request permissions, contact the publisher at leonil@outlook.com

Leonil Longno
Panabo City, Philippines
leonil@outlook.com

Preface

Thank you! for choosing **Structural Modeling and Design using RFEM, Classroom in a Book**

This book was written to help you begin exploring and mastering the essentials on how to do structural modelling, analysis and design using RFEM. Dlubal has been in the forefront of providing a masterpiece software in structural engineering and it is my hope that as you read and practice the exercises and problems provided in this book, you would begin to explore and use it accordingly in your structural models.

As a practicing structural engineer, I have tried to be concise and straightforward in my workflow and explanations in this book. This book uses the approach of learning by doing practice exercises. It however does not discuss every nook and corner of the software and should be studied side-by-side with the program's User Manual for its finer details and definitions.

At first look you might find the RFEM interface overwhelming, but that is a given in studying anything new. However, as you progress through the exercises and problems provided which goes from the simple to the more complex, you will then develop the confidence and skill on how to use it and appreciate its usefulness to whatever school thesis or commercial projects you undertake later.

I am happy to hear any feedback and look forward to continue improving this book and subsequent texts through your feedback. Feel free to contact me at leonil@outlook.com with any thoughts you would like to share, or better join the RFEM group available in Facebook.

I have been using RFEM in practice for some of my work. It is my hope that you become adept in its use and eventually help you become efficient in your work. All the best to you as you learn Dlubal RFEM, may you 'live long and prosper' using it.

LEONIL LONGNO
MScCE MBA CEng MIEI

Table of Contents

Acknowledgement

I would like to show my appreciation to several professional engineers and friends of mine who have reviewed the draft of this book in eagerness, and made suggestions for improvement.

Wenceslao Guieb, *Senior Structural Engineer, Road and Tunnelling Works;* his wife Lisa Guieb, *Associate Professor, PhD Candidate*; Asisclo Villafuerte, *Principal Structural Engineer for High-Rise Buildings*; Benedict Bayaca, *Principal Structural Engineer for High-Rise Buildings*; Deovan Luke B. Lamzon, *Project Engineer*

Lastly this book is dedicated to all members of the PGSE Facebook group, their camaraderie and exchanging of opinions through the years.

Introduction

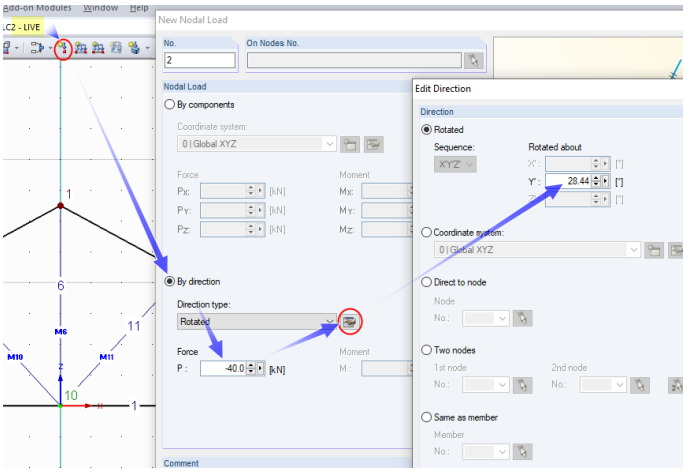
Dlubal RFEM is a general-purpose structural engineering, analysis and design software. Current statistics from the Dlubal website tells us it is used by more than 86,000 users globally in 95 countries. It can model and analyse just about any structure you can think of.

Dlubal is a software company exclusively dedicated to creating software for structural engineers. Their main headquarters is in Germany, but they have established several offices around the globe. Their products have been around for more than 30 years, and RFEM is their flagship software. The official website of Dlubal and their products is https://www.dlubal.com/

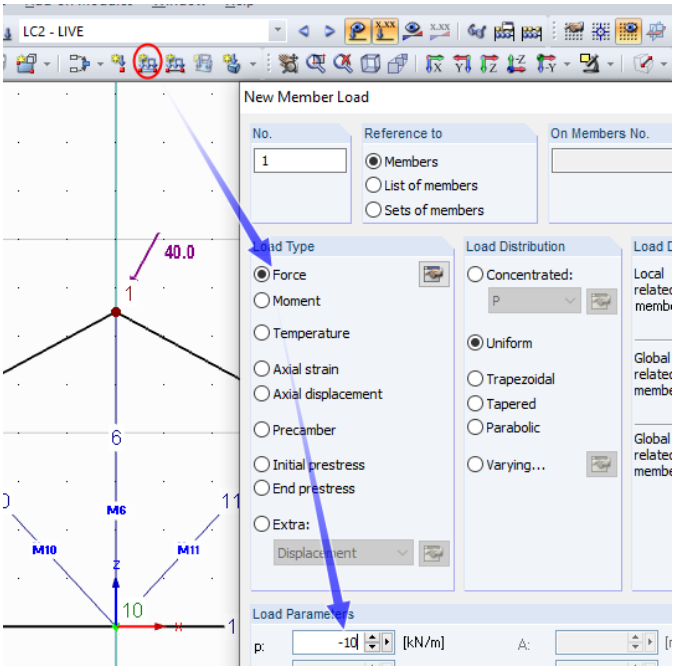
They have 2 main products, RFEM and RSTAB. RSTAB caters only with 2D or 3D structural frame member systems, while RFEM covers both frame and shell members. This book covers discussion and working on RFEM exclusively, of which RSTAB features are already covered and contained within it.

RFEM uses a modular approach. The main program RFEM is used to model and analyse structures. The post processing of the analysis results is separate and handled by 'add-ons' specific to how you want to process your results. There is an add-on for design of concrete members based on a specific code (e.g., ACI or EN1992), another for design of steel, another for glass, etc. There are add-ons for specialty analysis like dynamic response spectrum. And there are also add-ons that is used to ease and enhanced the modelling of structures and loads. For example, there is an add-on to automate the modelling of steel towers, and an add-on to automate wind loads on buildings according to a specific code like ASCE.

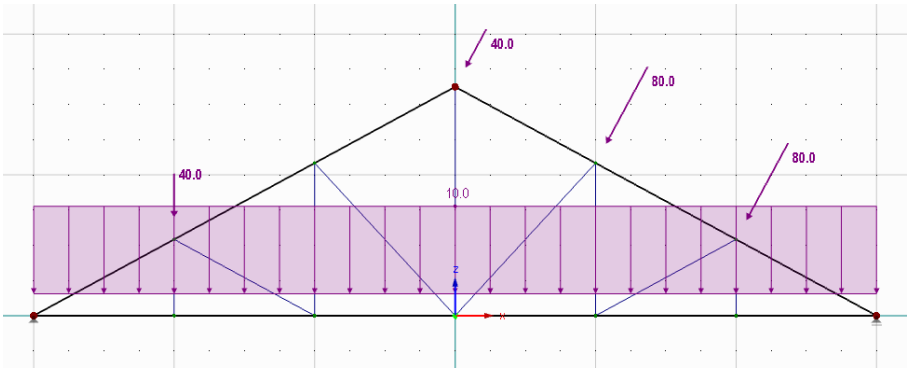
It supports analysis using materials made of concrete, metal, timber, glass, foil, gas and soil as shown. It includes an extensive list of country specific built-in libraries of sections, for example, on steel sections as shown below.



Add the uniformly distributed load (UDL) at the bottom chord as follows, then LC on the bottom chord member to apply the UDL of 10kn/m. Press ESC key to exit.

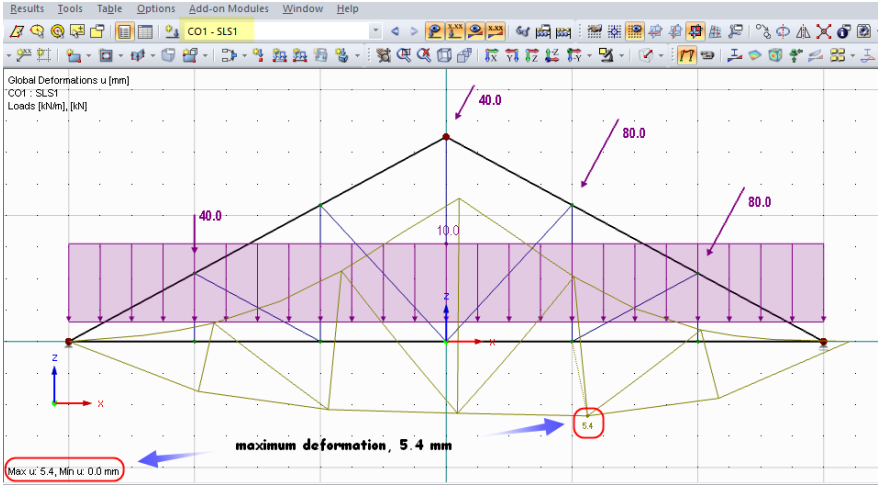


Your model with applied loads should look like the image below



Run the analysis, LC the “Calculate ALL” icon, or you can go to the menu Calculate>Calculate All

It will show you the deformation diagram by default. Change the load combination to SLS1 and it will show you the deformation due to combined dead weight and live load as shown below



From the Navigator pane we can display the Axial, Shear, and Moment forces diagrams as shown below.

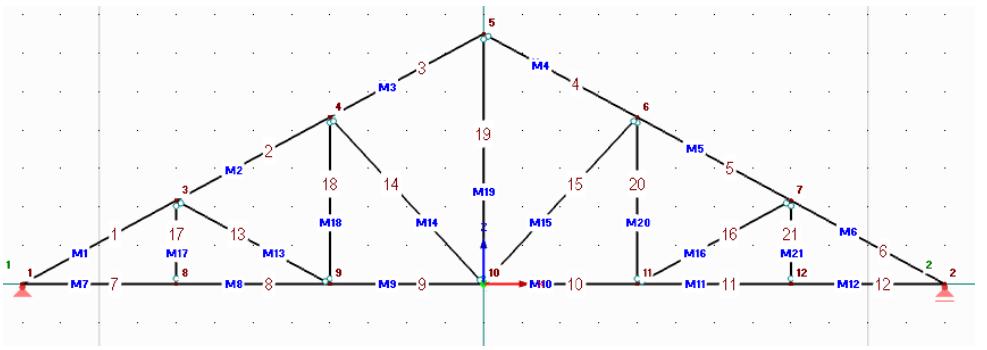
* notice that on cross-sections when you place your cursor on a table row, the concerned members will be color red telling you those are the members assigned to that row.

Your truss model will be generated in the workspace area as shown. Notice that unlike the model approach in Exercise 2, both the top and bottom chords have been separated into several members instead of just one member. However, by default the 'member type' of the top and bottom chord are 'Beam' not 'Truss', and notice also that the ends of these 'beams' are pinned (or released) at the ends. They are actually the same since a Truss type is a Beam type with released ends.

* to check the 'member type' on any member, DLC on that member

The diagonals and vertical are also of type Beams with released ends (see the small cyan coloured dots on its ends), not necessarily the same type "Truss (N Only)" we used in Exercise 2. However, the shear and moments produced on the web members will not be of significant amount and can be neglected. But if you prefer, you can easily change the member to 'Truss N Only' by selecting those members, RC and choose 'Edit Members' then change type from Beam to 'Truss N Only'.

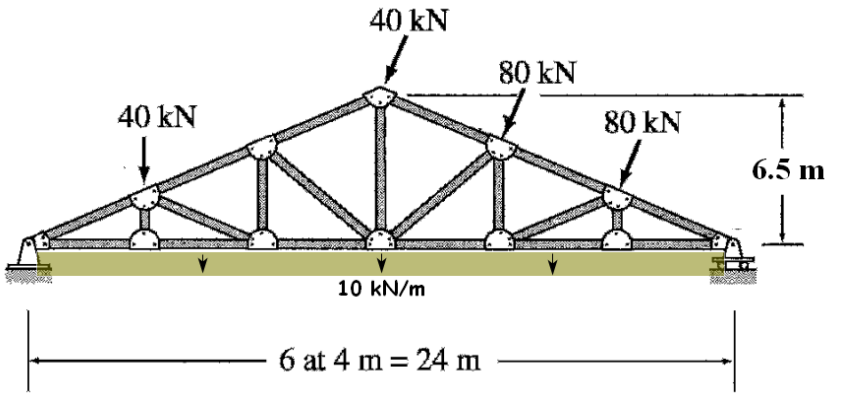
* to select multiple members, CTRL+Hold then LC on the members



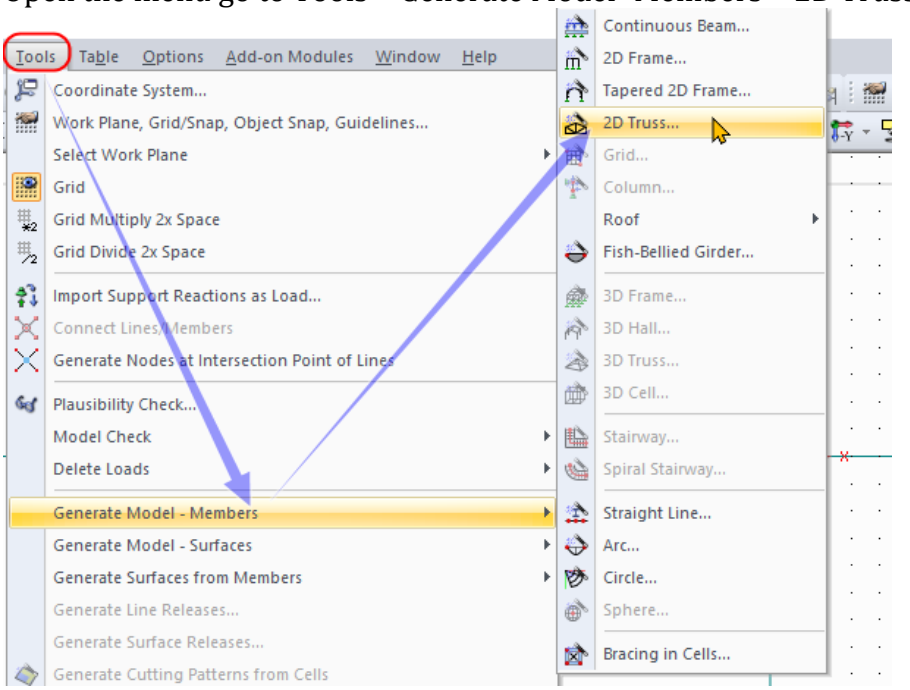
You can then add supports, load cases/combinations and loadings accordingly like in Exercise 2 to complete your model.

Exercise 4 – Truss using Model Generator

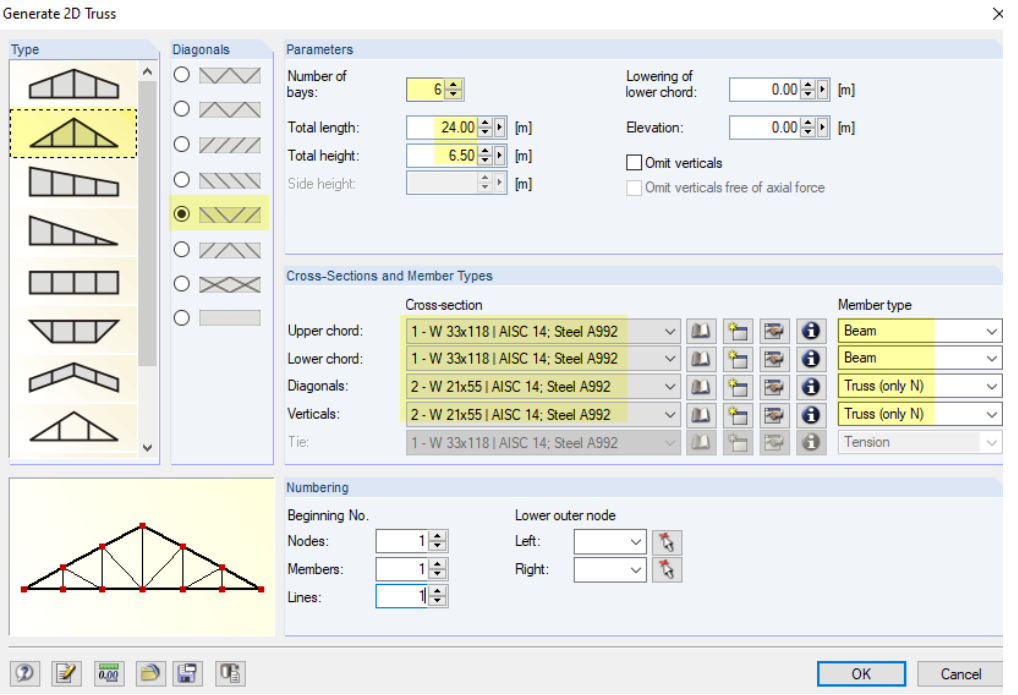
Model the truss in Exercise 2 using **Generate Model**



Open the menu go to Tools > Generate Model- Members > 2D Truss



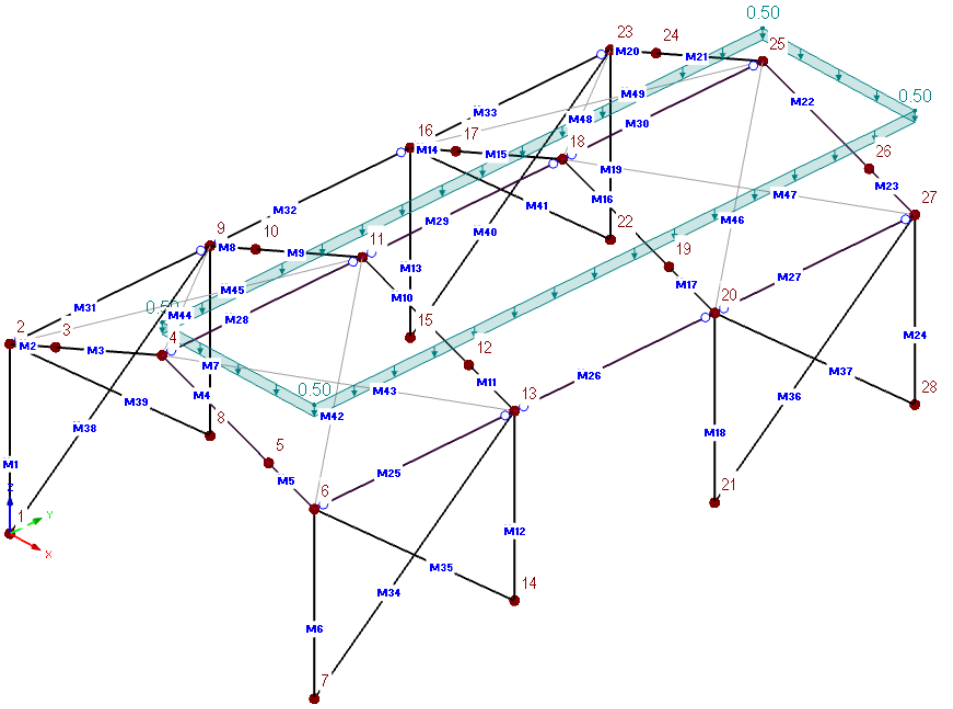
On the Dialog box, choose the Truss type and enter data as shown,



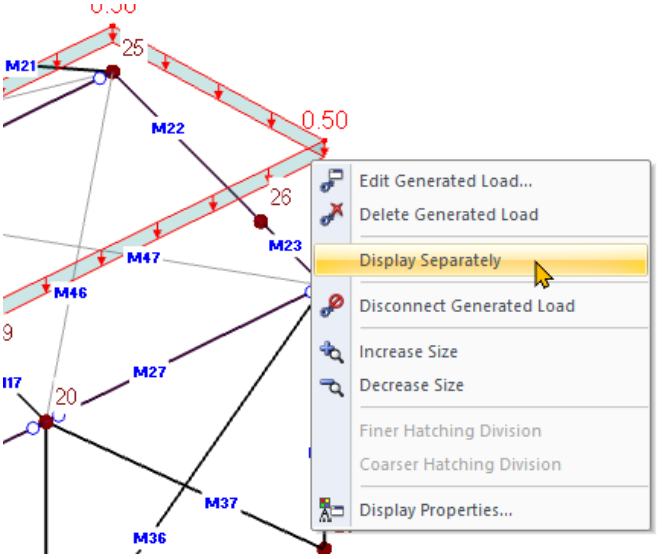
LC 'Ok' to exit.

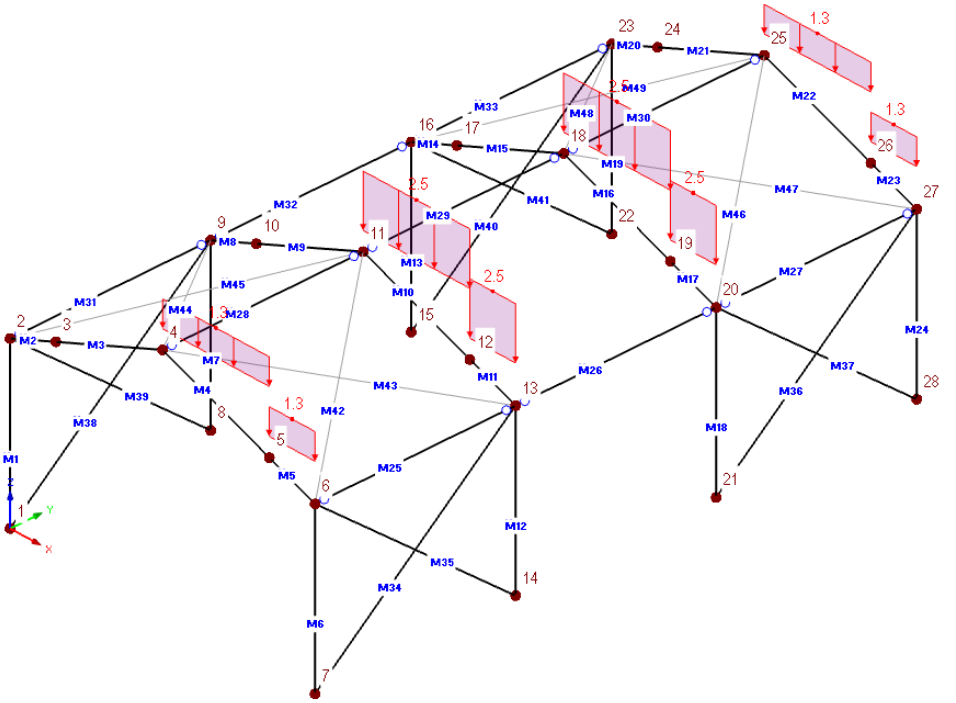
Your truss model will be generated in the workspace area as shown below. Notice that unlike our manually made model in Exercise 2, both the top and bottom chords have been separated into several members instead of just one member. Since we were given the option to define the 'member type' (not available when using the Blocks method), we were able to define both the top and bottom chord as 'beams' and the web members as 'Truss N Only'. In this case we only need to release the ends of the top chords at the apex to make it similar to Exercise 2.

To easily release those beam ends meeting at the apex, go to the Tables>Members tab then change the Hinge no of members 3 (end) and 4 (start) to a value of 1 as shown below



If you want to check the actual line load distribution of the area load, LC on the loading diagram (it will be highlighted in red color), then RC, then choose 'Display Separately'. It will show you similar to below:





From the image above, our intended one-way distribution of the area load to the rafters only was made possible by defining the excluded members that will not take any load (the tie beams and bracings).

(You can re-select the Load, RC then choose 'Display Separately' to remove the check mark and toggle it off.)

Next is to **change the active load case to Roofing System**. Our roofing system is an area load of 1.0 N/m2

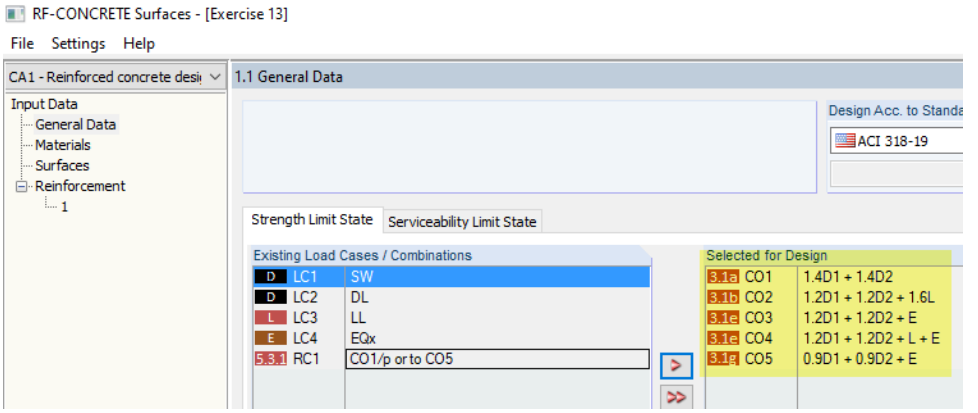
With similar action, go to Tools>Generate Loads>Form Area Loads on Members vis Plane

Notice that your previous selection is still active. So, in this case you only need to change the direction and amount of load as shown. LC 'ok' to exit.

Design of Slabs

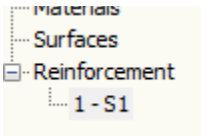
The workflow for the slabs is similar to all surface elements like walls and mat foundations. DLC on the add-on **RF-Concrete Surfaces.**

On General Data, select the Strength design load combinations and move them to the right box



Materials and Surfaces tree remains unchanged.

On the Reinforcement tree, since we only have 1 surface element, we will name it as S1



Note that if you have a big structure with walls, slabs and foundations, your list on reinforcement might expand to represent all those different surface elements. For example, your list might look something like the one shown below.

LEONIL LONGNO, ***MScCE MBA CEng MIEI***

is a Civil-Structural engineer for more than 30 years of industry practice both locally inthe Philippines and abroad. Completed both his graduate studies in the UP Diliman, he is also a PRC Registered Engineer, a Chartered Engineer in Ireland, and a member of IStructE UK, ASEP, PICE, SEAD and ACECOMS.